Mesh immersion technique for 3D moving domain calculation and applications to twin-screw extrusion and mixing

***R. Valette*, B. Vergnes and T. Coupez**

CEMEF - Ecole des Mines de Paris - UMR-CNRS 7635
B.P. 207 06904 Sophia Antipolis, France

Abstract

This work is concerned with the development of numerical techniques devoted to the simulation of the flow of a polymer melt in mixing processes such as twin-screw extrusion. In mixing or twin extrusion process simulation, the absence of symmetry of the moving boundaries (the screws) implies that their rigid body motion has to be taken into account by using a special treatment. In this study, we introduce a new technique called Mesh Immersion Technique (MIT), which consists in : a) using a P1+/P1-based (MINI-element) mixed finite element method for solving the velocity-pressure problem, b) solving the problem in the whole barrel cavity and imposing a rigid motion (rotation) to nodes found located inside the so called immersed domains. Each sub-domain (screw) is represented by a surface mesh. The independent meshes are immersed into a unique background computational mesh by calculating the P1 approximation of the function giving the distance to their respective surfaces. A multiphase approach, combined with parallel computing, is used to compute the flow of generalized Newtonian fluids in a complex system such as a twin screw extruder or a batch mixer, including moving free surfaces.

1 Fluid domain

Let us consider the flow of an incompressible fluid in a domain $\Omega(t)$ representing, for example, the domain occupied by the fluid inside a portion of a twin screw extruder. The velocity-pressure (u,p) problem is governed by mass and momentum equations (where τ is the extra-stress tensor) :

$$\begin{aligned} &-\nabla p + \nabla.\tau = 0 \\ &\nabla.u = 0 \end{aligned} \qquad (1)$$

2 Boundary conditions

No slip boundary conditions on the moving screws and on the barrel walls are imposed. Solving this problem by discretizing the domain occupied by the fluid would imply mesh distortion and dynamic remeshing [1]. To avoid this, one uses a fixed Eulerian computational domain which contains the “domain” occupied by the moving boundaries, i.e. the screws. Equations (1) are then solved using a multiphase approach in a mixed finite element method [2].

3 Mesh immersion technique and multiphase modelling

To identify the position of the screw/fluid interfaces, one uses a P1 approximation of the distance to these interfaces. Solid body motion is accounted for by directly assigning velocities to nodes of the grid (as for usual boundary conditions) belonging to elements located inside the screws [3]. In boundary elements, a mix rule on viscosities is used as in [2] to obtain a homogeneous viscosity. The same method is used to compute a free surface flow, using a level set approach for the identification of the surface position [2] at each time step.

4 Numerical results

The preceding method has been successfully applied to some industrial configurations. For example, figure 1 shows the pressure field in a combination of left- and right-handed screw elements (twin screw extrusion). Figure 2 shows the interface position in a batch mixing application.

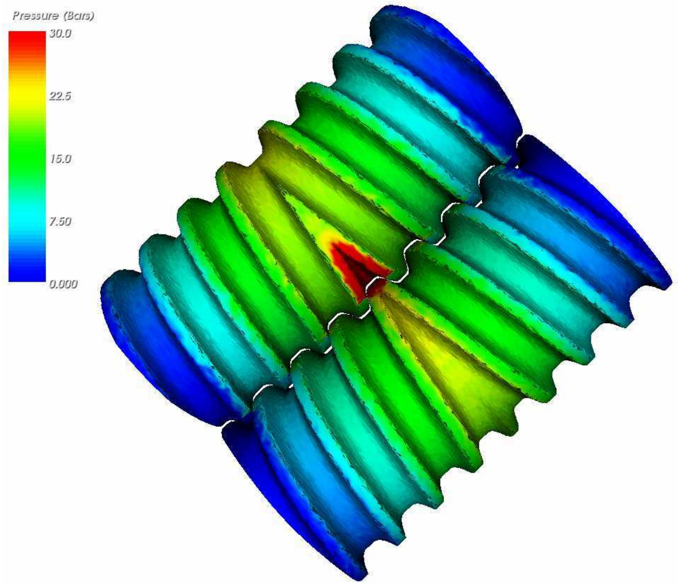

Figure 1: Pressure field on the screws for a combination of left- and right-handed screw elements (twin screw extrusion).

Figure 2: Free surface position in a batch mixer.

5 References

1. C. W. Hirt, A. A. Amsden and J. L. Cook, *An arbitrary Lagrangian-Eulerian computing method for all flow speeds*, J. Comp. Physics, **14**, 227 (1974).
2. O. Basset, H. Digonnet, H. Guillard and T. Coupez, *Multiphase flow calculation with interface tracking coupled solutions*, Proc. Int. Conf. on Computational Methods for Coupled Problems in Science and Engineering, Barcelona (2005).
3. J. S. Raut, V. M. Naik and T. R. Jongen, *Efficient simulation of time-dependent flows: application to a twin screw extruder*, A.I.Ch.E. J., **49**, 1933-1946 (2003).